International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 8, Issue 10 (October 2013), PP. 01-08

Optimization of Plate Forming Process Using Ansys Parametric Design Language

Sharangouda Patil[1], Prof. Bharath S Kodli[2]
[1]*PDA College of Engg. Gulbarga,*
[2]*Tejavi Techno Solution Bangalore.*

Abstract:- Forming operation plays important role in manufacturing industry. Forming has the inherent advantage of faster production rate with uniform to variable thickness formation. The objective of this work is to obtain an optimal dimensions in the consideration of the influence of the metal flow deformation in Sheet metal forming process. Finite element method in conjunction with optimization algorithm(APDL) was used to analyze the effect of sheet metal size, fillet size and depth of forming on forming load in axisymmetric finite element process. Finite element software (ANSYS) was used to Simulate forming process and then performing a series of optimization iterations in order to obtain the optimal size and shape of punch and die geometry with reference to punch load. The material used is aluminum metal matrix composite (AlMgSi matrix with 15% SiC particles).

Keywords:- Plate Forging, Finite element method, Optimization

I. INTRODUCTION

The finite element method is a numerical procedure that can be applied to obtain approximate solutions to a variety of problems in engineering. Steady, transient, linear, or nonlinear problems in stress analysis, heat transfer, fluid flow, and electromagnetism problems may be analyzed with the finite element method the idea of representing a given domain as a collection of discrete parts is not unique to the finite element method. It is very important to study the deformation behaviour of these materials to produce defect-free products.

Optimization of forming process design and forming process plan for various work materials can be based on the maximization of production rate. rninimization of production cost. minimization of die cost. maximization of product quality. The main factors effecting sheet and plate metal forming are die shape (angle, bend radii), sheet or plate thickness, material properties and frictional condition at the plate/die interface ANSYS Parametric Design Language (APDL) is a scripting language that can be used to build the model in terms of variables. APDL is used to build the model parametrically to enable variables changes during the optimization process, so that the best combination of design parameters is obtained for the specified objective function, where in the present study, the objective function is the load required for the forging process.

Design Variables (DV) are the plate thickness, bend radius and the bend angle shown in Figure 1. The equivalent-, strain and contact gap are defined as State Variables (SV). I The state variable is working as constrain in the' optimization process, forcing the design parameters to be adjusted in order to keep the strain and the contact gap below specified values, to avoid wrinkles and necking. In ANSYS Optimizer, the objective function must be a state variable to be minimized. (State variable is a variable that is changing during the process depending on a design variable

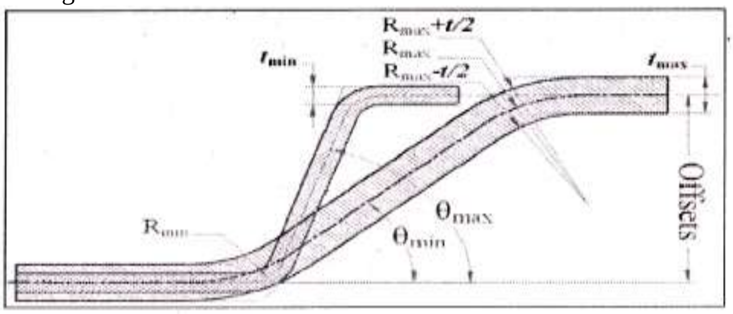

Figure 1: Geometrical parameters (Design Variables).

II. ALUMINIUM METAL MATRIX COMPOSITES

Aluminium is the most popular matrix for metal matrix composites (MMCs)-Aluminium alloys are attractive due to their low density, their capability to be strengthened by precipitation, their good corrosion

resistance. high thermal and electric conductivity, and high damping capacity) ' Aluminium matrix composites (AMCs) offer a large variety of mechanical properties depending gn the chemical composition of the aluminium matrix. They are usually reinforced by continuous and discontinuous reinforcements. Discontinuous reinforced AMCs are very altercative for their' isotropic mechanical, properties (higher than their unreinforced alloys) and their low costs (low prices of some of the discontinuous reinforcement such as SiC particles or Al2O3 whiskers).

Figure 2 shows particulate and whiskers reinforcements. both are considered discontinues reinforcements.

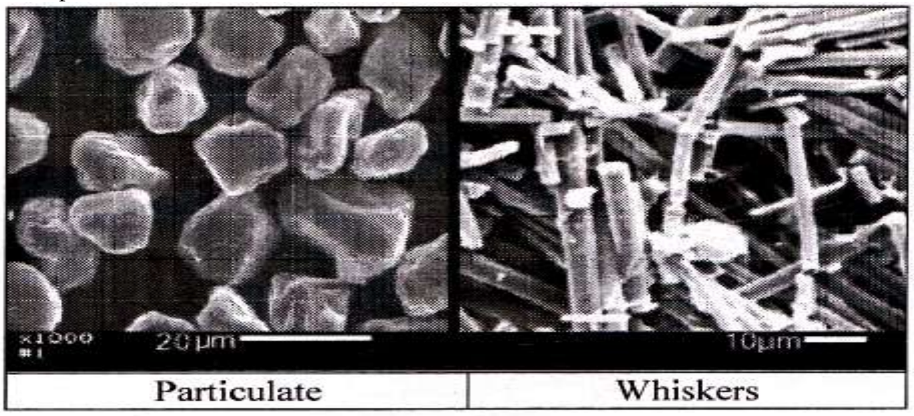

Figure 2: Discontinuous reinforcement

MMCs cause particles and whiskers breakage, and normally result in cracks at the outer surface of the work material. To avoid particles breakage which leads to cracks, the equivalent strain of the material must be kept at low values.

III. FINITEELEMENT MODEL

A circular plate made of aluminium metal matrix reinforced with 15 volume % silicon carbide particle is going to be formed as shown in Figure 3. The metal plate is represented with a rectangle (a, b, c and d) and its thickness is line (b-c). The upper and lower dies are represented with lines having two geometrical parameters (Design Variables), the bending angle and the bending radius (+ or - half plate thickness depending on the bend direction). The plate is represented with geometrical model consisting of assembly of finite element. Equations relating the distribution of forces and displacements of the metal are established and the boundary condition and die movement are imposed.

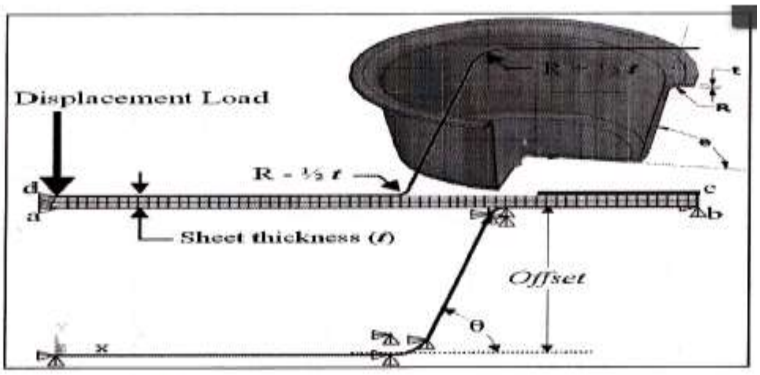

Figure 3: The discretised model showing boundary condition

The cornponents of the model are shown in Figure 3. The Metal plate. is made of AlMgSi reinforced with 15% SiC particles and has an initial thickness of 3 mm. Just half of the plate and the die is considered for the analysis to reduce computational time and cost.

Table l: Isotropic Hardening characteristics of Plate material

Strain	**Stress**
0.003428	240
0.1	280

0.2	300
0.4	320
0.5	321
0.6	325
0.8	330

TABLE 1 : MATERIAL DATA

Three types of elements are used in the model. The plate is built up of two dirnensional 4-node viscoplastic solid elements (ANSYS type VlSCOl06). A rigid to flexible contact pair is used to represents die/plate contact. A two dimensional 2-node surface-to-surface contact element (ANSYS type CONTA l72) is used to represent friction and sliding contact for the deformable surface of plate while a two dimensional target element (ANSYS type TARGEI69) is used to model the rigid surface of the upper and lower die.

The axis of symmetry (line a-d) of the plate is constrained in (X) direction. The lower die is constrained in (X) and (Y) direction. A displacement load is applied to the target element (TAR.GEI69) associated to the lines representing the upper die profile in negative (Y) direction.

IV. OPTNTZETION PROCESS

Optimization is a popular subject in finite element analysis, and is becoming more important goal in the product development process analysis. This trend is facilitated by the ever-increasing computing power used to solve analysis problems. For the design engineer, it is often the real end goal.

BASIC CONCEPT OF OPTIMIZATION

Optimization is quite an interesting aspect of engineering practice that cuts across all branches of engineering. In the production sector, for example, the reduction of material (Figure 4) used in manufacturing is possible when optimization is incorporated beforehand.

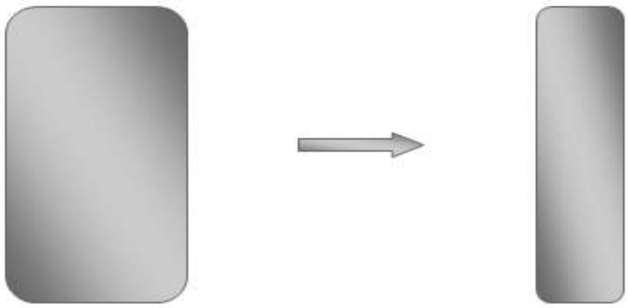

FIGURE 4: MATERIAL REDUCTION

Definition of Optimization

Optimization can be defined as the process of finding the conditions that give maximum or minimum value of a ‘function’. Where effort required or benefit desired for a given practical situation is expressed as a ‘function’ of certain design variables. This is illustrated in the Figure 5.

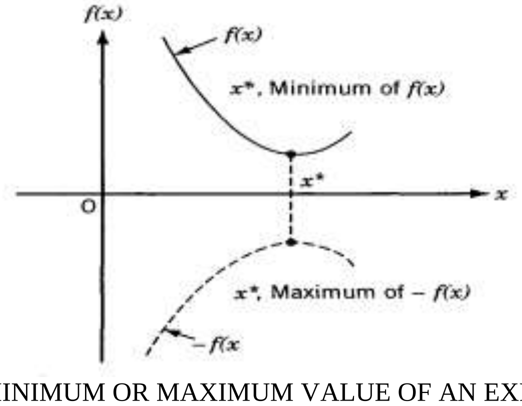

FIGURE 5: MINIMUM OR MAXIMUM VALUE OF AN EXPRESSION

The goal of the optimization process is to find the best solution for the given problem in the design space defined to the optimization algorithm. Optimization model consists of three components: design variable (independent variable), constraints (state variables or dependent variables) and the objective function (dependent variable to be minimized). The optimization method used in the current study is called Sub-problem approximation method. The method can be described as an advanced zero-order: method which requires only the values of the dependent variables, and not their derivatives. To start the Optimization process, parameters are first defined. They are referred as the design set. These parameters include design variables, state variables and the model objective. Their values are modified throughout the Optimization process. Design variables are independent quantities that are constrained within a specified range and are changed during the Optimization analysis process.

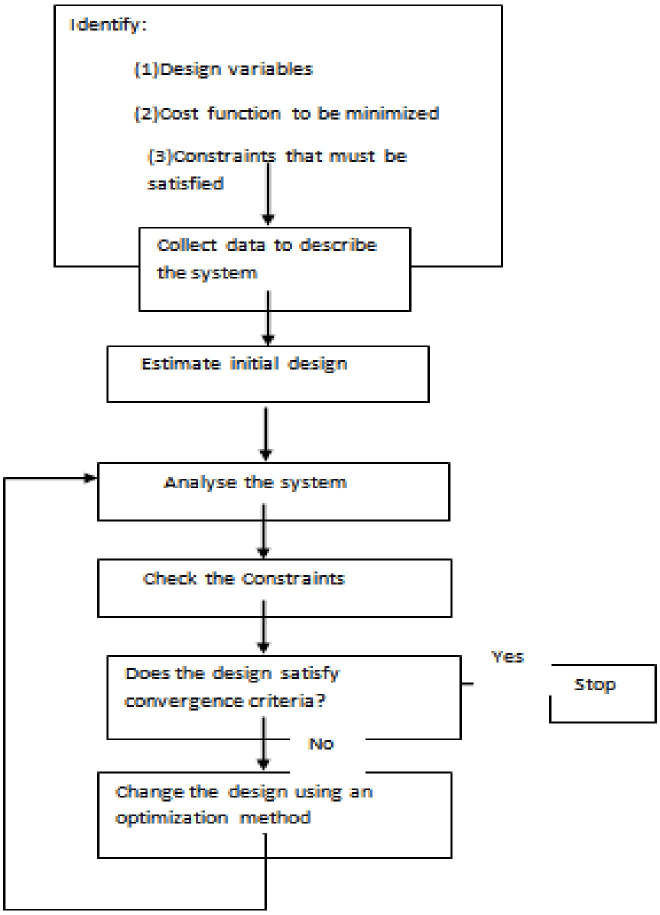

FIGURE 6: THE OPTIMIZATION PROCESS

OBJECTIVE FTINCTION

The variable that is going to be defined as an objective function must be a state variable that is changing during the process such as stress or strain or forming load. The objectives is the load required for the forging process is taken as the objective function. Here the main work is to limit forging load in the process by design optimizing the process for design variable and state variable.

V. RESULTS AND DISCUSSION

The analysis is carried out for the initial geometry to find deformation, stress and contact pressure and possible plastic strains. Large deformations are activated to capture the penetrations and plastic flows along with contact simulations. Using commercial FE package (ANSYS). the FE model was build parametrically for plate metal forming using APDL in order to enable ANSYS optimizer carry out a few optimization iteration by changing the design 'variables (parameters) Table 2 shows ten design sets carried out by the optimization algorithm' The set eight is the optimal one.

R1 –Fillet Radius T1: Thickness of the Sheet Metal
TH– Angle H1- Sheet Metal depth of forming

		SET 1 (FEASIBLE)	SET 2 (FEASIBLE)	SET 3 (FEASIBLE)	SET 5 (FEASIBLE)
MPLASTIC	(SV)	0.93112	0.77276	0.82272	0.76642
T1	(DV)	4.5537	4.4651	3.9233	2.6210
R1	(DV)	3.8284	4.7431	5.2254	6.7564
TH1	(DV)	82.233	76.984	81.123	80.478
F1	(OBJ)	0.13848E+07	0.22139E+07	0.15607E+07	0.48217E+07

		SET 8 (FEASIBLE)	SET 9 (FEASIBLE)	SET 10 (FEASIBLE)
MPLASTIC	(SV)	1.1602	0.97024	1.1679
T1	(DV)	4.5498	4.4898	4.5586
R1	(DV)	3.7112	3.7342	3.7187
TH1	(DV)	82.239	82.781	82.219
F1	(OBJ)	0.12313E+07	0.15653E+07	0.13232E+07

Table 2

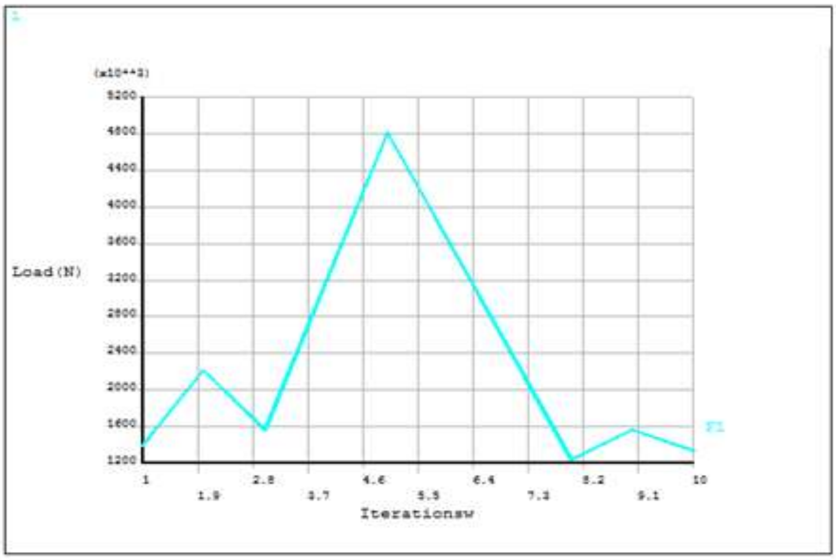

Fig 7 : iterations Vs Punch load

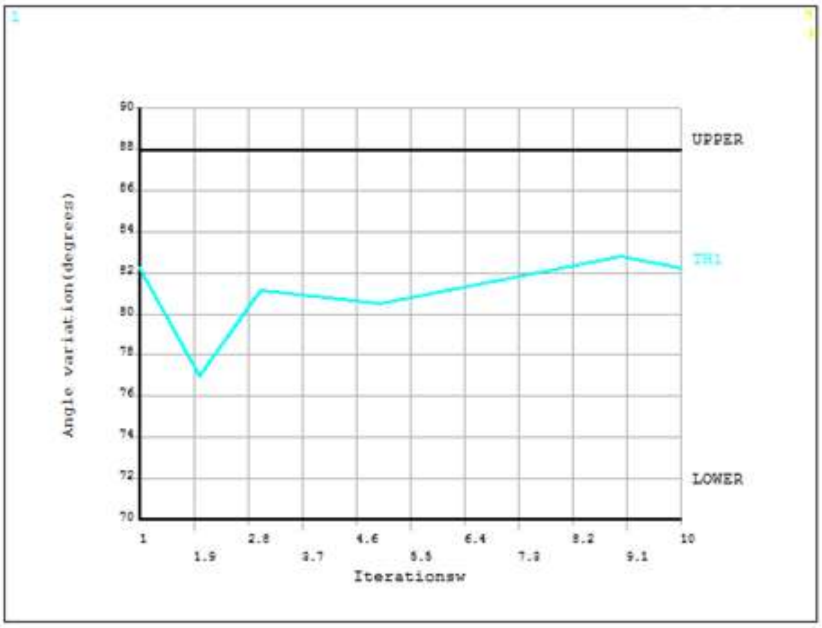

Fig 8 : Iterations Vs Thickness Variation

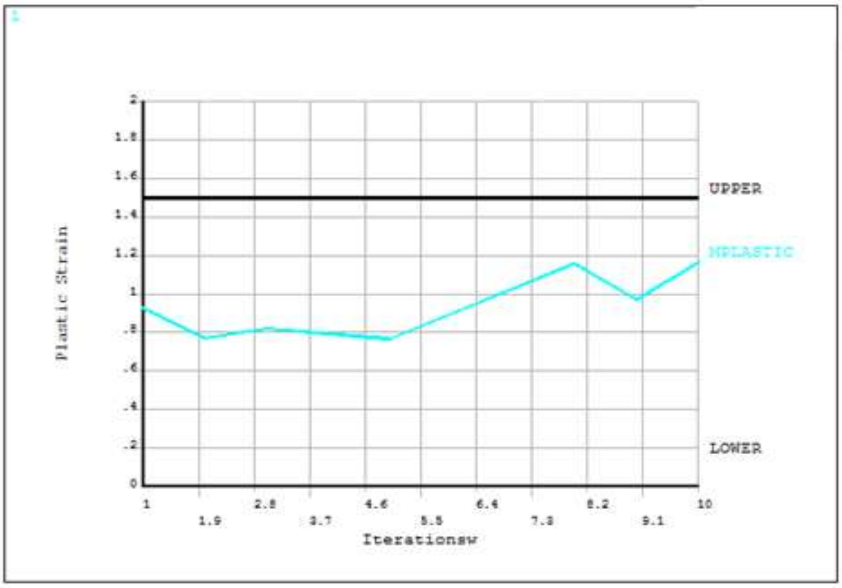

Fig 9 : Iterations Vs Plastic Strain

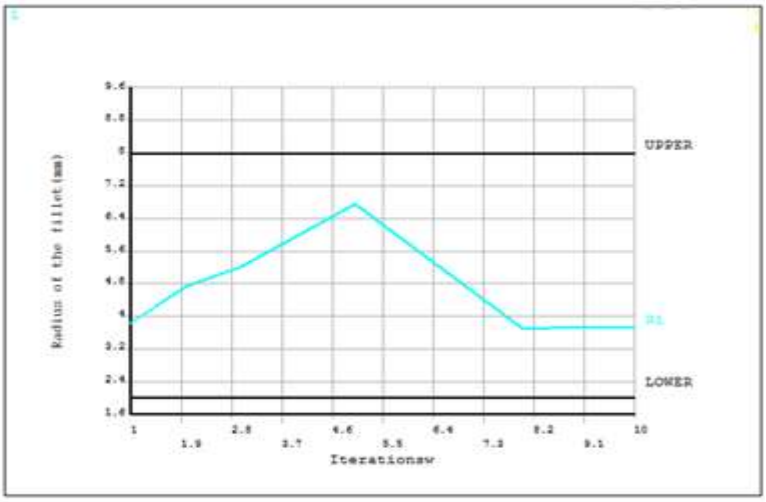

Fig 10 : Iterations Vs Fillet Variation

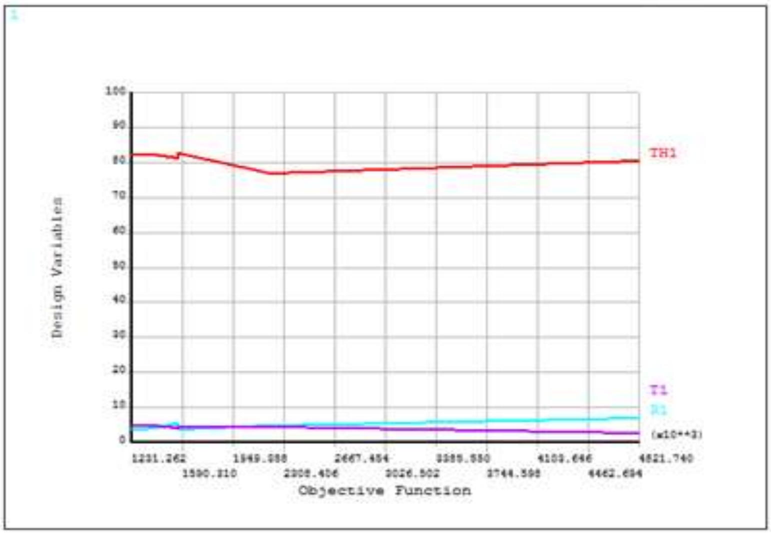

Fig 11: Objective function to Design variables

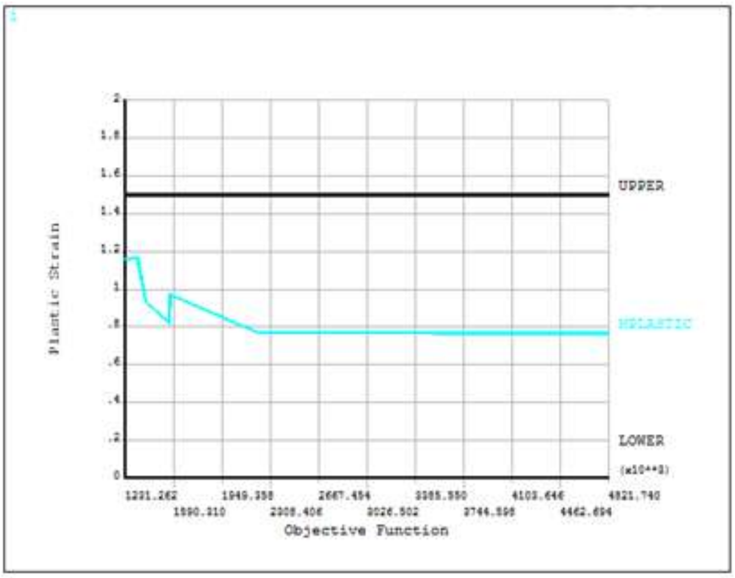

Fig 12: Objective Function to Plastic Strain

Maximum displacement of 28mm can be observed in the Sheet metal process. The sheet metal is formed to the required shape by the punch movement. Maximum displacement can be observed at the inner regions and minimum displacements in the outer regions.

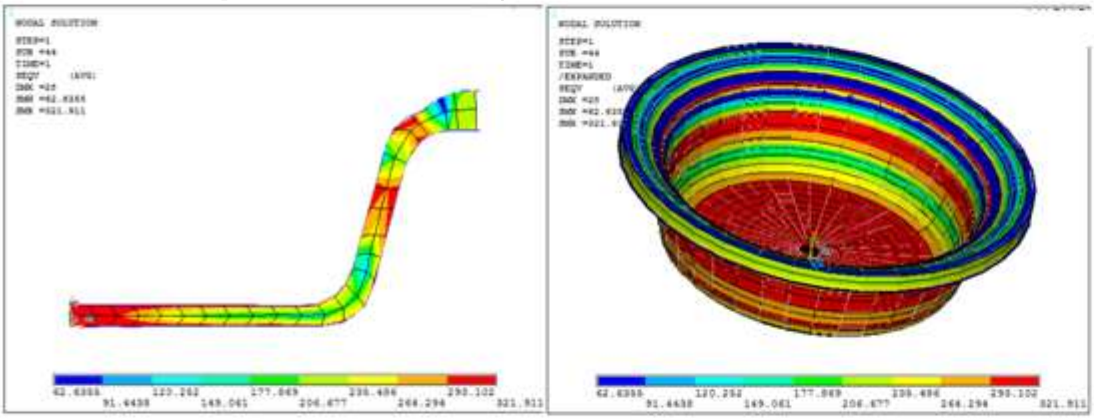

FIG 13 : VONMISES STRESS RESULTS FOR INITIAL STRUCTURE

VI. CONCLUSION & FURTHER SCOPE

- Sheet metal forming process is simulated through APDL code. The code helps in obtaining number of design sets giving possible deformations, stresses, plastic strains etc. Depends on requirements, a design set can be choosen for the particular application. Also the complication in finding the load required for forming process can be easily estimated and saves the designer time. From the results it is observed that sheet metal inclination, thickness and fillet plays major role in deciding the safety and error free formed products.

FURTHER SCOPE:

- Analysis process can be carried out with thermal effects.
- Problem can be executed in three dimensional space
- Impact analysis can be carried out to find behaviour of the members in contact.
- Topology optimisation can be carried out to find optimum thickness required for punch, die and blank holders
- Composite usage can be checked.

BIBLIOGRAPHY

[1]. R.J.Duffin, E.L.Peterson and C.Zener, Geometric programming: theory and applications, Wiley, New york, 1967.

[2]. G.B.Dantzig Linear Programming and Extensions, Princeton University Press.

[4]. R. A. Gettatly and R. H. Gallagher, "A procedure for automated minimum weight structural design, Part

[5]. I - Theoretical basis, Part II - Applications," Aero. Quart. Part I, Vol. 17, pp. 216-230 and pp.332-342, 1966.

[6]. M.M. Denn, Optimisation by variational methods, McGraw-Hill , New York 1969.

[7]. G.S.G. Beveridge and R.S. Schechter, Optimisation : theory and practice, McGraw-Hill, New York , 1970.

[9]. J.L. Kuester and J.H. Mize, Optimisation techniques with FORTRAN, McGraw-Hill, New York, 1973.

[10]. M.J. Panik, Classical Optimisation: foundations and extensions, North-Holland Publishing Co., Amsterdam, 1976.

[11]. D. Koo, " Elements of Optimisation" Springer-Verlag, New York, 1977.

[12]. D.G. Carmichael, Structural modeling and Optimisation, Ellis Horwood Chichester, 1981.

[13]. Morris,A.J**.** Foundations of structural optimization: a unified approach. John Wiley & Sons, 1st ed., UK, **1982**

[14]. Y. M. Xie and G. P. Steven, "A simple evolutionary procedure for structural Optimisation," Comp.Struct. Vol. 49, pp. 885-896, 1993.

[15]. S.S. Rao – Optimization, Theory and Applications, Wiley inter-science, 1996

[16]. Ernst Hustedt, AMES Ltd. 1999, (Air New Zealand Engineering).

[17]. Jim Patterson (Hendrickson Trailer Suspension Systems). 2000.

[18]. Finite Element Procedures – Klaus-Jurgen Bathe, Prentice Hall of India Pvt. Ltd.- Sixth Edition 2002.

[19]. Introduction to the Finite Element Method, Desai/Abel – CBS publishers 2002.

[20]. Finite Element Analysis – C.S. Krishnamoorthy, Second Edn, TMH – 2002.

[21]. Hursha Narayan (Robert Bosch Corporation). 2002

[22]. Joe Metrisin (Florida Turbine Technologies, Inc.). 2002

[23]. Concepts and Application of Finite Element Analysis Rober D. Cook, David S – Joh Wiely & Sons Pte. Ltd. Fourth Edition 2003.

[24]. Finite Elements in Engineering – Tirupathi R. Chandrupatla, Ashok D. Belegundu, Prentice – Hall of India Pvt. Ltd, 2003.

[25]. Arrora, J. "Introduction to Optimum design", 2nd ed., Academic Press, 2004